Defense Technical Information Center Compilation Part Notice

ADP014102

TITLE: A CFD Coupled Acoustics Approach for the Prediction of Coaxial Jet Noise

DISTRIBUTION: Approved for public release, distribution unlimited
Availability: Hard copy only.

This paper is part of the following report:

TITLE: Aging Mechanisms and Control. Symposium Part A - Developments in Computational Aero- and Hydro-Acoustics. Symposium Part B - Monitoring and Management of Gas Turbine Fleets for Extended Life and Reduced Costs [Les mecanismes vieillissants et le controle] [Symposium Partie A - Developpements dans le domaine de l'aeroacoustique et I'hydroacoustique numeriques] [Symposium Partie B ...

To order the complete compilation report, use: ADA415749

The component part is provided here to allow users access to individually authored sections of proceedings, annals, symposia, etc. However, the component should be considered within the context of the overall compilation report and not as a stand-alone technical report.

The following component part numbers comprise the compilation report:
ADP014092 thru ADP014141

A CFD Coupled Acoustics Approach for the Prediction of Coaxial Jet Noise

G.J. Page, J.J. McGuirk, P. Behrouzi, M. Hossain
Department of Aeronautical and Automotive Engineering
Loughborough University
Loughborough, Leics. LE11 3TU, UK

M.J. Fisher
ISVR, University of Southampton
Highfield, Hants. SO17 1BJ, UK

SUMMARY

Prediction of jet noise is important for civil aircraft. Some CAA methodologies predict the full unsteady flow field of a jet in order to ascertain the far-field noise. The approach adopted here is to utilise CFD to obtain steady state information using a turbulence model and hence to provide inputs to a semi-empirical noise model, herein after referred to as the four source model. Predictions of a coaxial jet in comparison to laser measurements show that the CFD methodology can reproduce the experimental velocity field mixing and turbulence intensities. This leads to confidence that the CFD model can predict the influence of geometrical changes (such as nozzle area ratio) on the mean and turbulence field and so increase the validity of the four source model. Predictions of two geometries with differing area ratios showed that peak turbulence intensities are increased in the smaller area ratio, but this can be accounted for by the use of a 'fully mixed' velocity in a four source model for jet acoustics. Predictions of a three-quarter cowl geometry were used to determine the equivalent parallel coaxial jet found immediately downstream of the bullet. This was achieved by integrating the areas and mass flows in the primary and secondary streams at the nozzle exits and downstream of the bullet. It is found that a velocity ratio of 0.7 and area ratio of 2.6 at the nozzle exit planes can be considered equivalent to a velocity ratio of 0.5 and area ratio 5 in the parallel flow downstream of the bullet. Input of such information from a RANS CFD prediction may be a relatively simple method for extending the applicability of the four source model.

1 INTRODUCTION

Jet noise is an important component of the noise emission of civil aircraft. Even for high bypass ratio engines, jet noise is the most prominent source at the full power take off condition. For low to medium bypass ratio engines, internal mixing of the core and bypass streams using a forced mixer offers significant jet noise reductions. At high bypass ratios, however, the noise benefit of internal mixing is small due to both the high flow area ratio and the high velocity ratio. In addition, a long bypass cowl has significant weight and drag penalty.

As a result of the above, nozzle designs are actively being sought which might result in significant jet noise reductions for high bypass ratio, separate jet exhaust configurations. A novel approach by which this might be achieved is to modify the complex coaxial jet flow development downstream of the nozzles, either by geometric or acoustic means. Recent model-scale exhaust tests involving fairly subtle changes to the nozzle profile have met with considerable success. By serrating the basically conical nozzle exit profiles, substantial jet noise reductions have been measured under both static and flight-simulation conditions [1]. In order to exploit the benefits of this type of nozzle design, an understanding of the flow and resultant noise production is required.

The most general methodology for the prediction of far-field jet noise is to compute the near-field unsteady flow-field using a DNS or LES technique in conjunction with an acoustic analogy. Initial work

Paper presented at the RTO AVT Symposium on "Ageing Mechanisms and Control: Part A – Developments in Computational Aero- and Hydro-Acoustics", held in Manchester, UK, 8-11 October 2001, and published in RTO-MP-079(I).

has concentrated on application to supersonic jets[2,3,4]; more recently subsonic jets have also been computed [5] with some success. Whilst this approach could be used for coaxial jets with serrated nozzles, many current LES methods would be unable to handle the complex geometric features and the large computer run times would make it infeasible for any design application. An alternative approach is to use CFD to predict the steady flow-field using a Reynolds Averaged Navier-Stokes (RANS) method with an appropriate turbulence model, and to use the predicted mean velocity and turbulence properties as inputs to an appropriate jet noise model [6,7] in order to predict the far-field. This type of approach has been used for coaxial jets with promising results[8]. Methods which attempt to resolve all (in DNS) or at least part (in LES) of the unsteady acoustic pressure within the flow field calculation may properly be considered as time-domain Computational Aeroacoustics (CAA). The approach of references [6,7,8] which only extract statistically averaged information from a steady state flow calculation for input to an acoustic model are here referred to as CFD Coupled Acoustics (CCA).

As an alternative to using an acoustic analogy basis for the noise model, a semi-empirical approach to jet noise modelling can be created by using a database of experimental jet noise measurements. These are used in combination with simple flow parameters extracted from experimental observations of mean flow and turbulence in coaxial jets; the best example of this approach is the 'four source model' [9,10]. Inherent in this type of approach are assumptions regarding parameters such as mean velocity and turbulence intensity profiles in the jet mixing layers and so changes in noise due to geometric changes in the nozzle design which affect the turbulence properties in the mixing layers of the jet cannot be easily predicted. The aim of the present work is to examine the benefit of adopting the CCA approach in order to improve the generality of the four source model. This will be done by using a RANS-based CFD prediction as a flow model for coaxial jets and extracting from these predictions the parameters required by the four source model (e.g. effective jet definition and turbulence intensity). This should make the four source model capable of extension to any configuration for which CFD analysis is possible.

Firstly, comparisons are shown between experiment and CFD for a coaxial, co-planar, nozzle representative of that used in the measurements of Ko and Kwan [11] in order to confirm the accuracy of the RANS-CFD approach adopted here, in particular the turbulence model. CFD predictions are then shown for a similar co-planar configuration with differing area ratios to show how this affects peak turbulence intensity and location. A three-quarter cowl geometry is also predicted in order to confirm how this affects turbulence levels and to establish a firm basis for translation of real nozzle exit conditions into an equivalent co-planar nozzle.

2 METHODOLOGY

2.1 Experimental

Experimental measurements were carried out in a water-tunnel facility designed specifically for near-field jet mixing problems [12]. The working section was 1.125m long, 0.37m wide and 0.3m high and was made of perspex to allow ample optical access. The primary and secondary flows have separate circuits with flow rates monitored by rotameters. It is also possible to provide a co-flowing stream outside of the jet nozzles to simulate a flight stream. Turbulence management units are provided in both jet flows in order to give a uniform profile with as low turbulence level as possible at the exit from the nozzle system. The tertiary flow is also used to create a very low velocity co-flow in order to stabilize the flow in the downstream mixing region.

An LDV system is used to provide mean velocity and turbulence data. This consisted of a single channel forward-scatter fringe-mode velocimeter with a Helium-Neon laser operating at a nominal power of 10mW. Naturally occurring particles within the water supply were found to be sufficient to produce a high data rate with no additional artificial seeding. Typical sampling rates were 1kHz and mean velocities were produced from a minimum of 25,000 samples. Bragg cell frequency shifting was incorporated into the system to provide sensitivity to flow direction and allow high turbulence intensities to be captured. Signal processing was carried out using a TSI IFA 550 frequency processor (as described in [13]). No corrections were made for sampling bias as errors were minimised by use of high data rates compared to typical velocity fluctuation frequencies, as suggested by Erdmann and Tropea [14].

2.2 Computational

The CFD methodology employed in the current work solves the time-dependent, mass-weighted and ensemble-averaged conservation equations for mass, momentum and total energy. The governing equations are discretized using a finite volume technique on a structured curvilinear multiblock mesh, using collocated variables. A segregated approach is used; each equation is discretized in the form of a scalar implicit equation with linearized coefficients. The pressure-correction technique is used to couple the velocity and pressure fields, and is implemented in such a manner to work efficiently for both incompressible and compressible flows [**15**].

The cell face flux is discretized using a basic first order upwind scheme for the convective flux (based upon the cell face normal velocity), and central differencing is used for all other terms. Higher order discretization schemes are implemented as a deferred correction to the first order upwinding of the convective part using a general upwind-biased MUSCL type scheme.

The standard k-ε turbulence model is used together with wall functions on the nozzle walls [**16**].

3 RESULTS

3.1 Calibration of Co-planar coaxial nozzle

The four source model [**9**] used the experimental data of Ko and Kwan [**11**] to justify that the peak turbulence intensity in the interaction region of a coaxial jet configuration should be 10% of the primary jet velocity, rather than the standard single jet turbulence level of 15%. The aim of the experimental measurements and initial CFD predictions in this work was to confirm the correctness of this assumption, and to validate the CFD model so that it can then be used to assess the influence of geometry changes on mean and turbulent quantities.

The geometry chosen for the experimental study is representative of that used by Ko and Kwan in that an identical secondary to primary area ratio (β) of 2.7 was used, whilst the contraction ratios, lip thicknesses, etc. were all somewhat different due to design considerations (see Fig. 1). The Reynolds number based upon primary diameter (D_i) and bulk average primary velocity is 80,000. The small geometrical differences were considered as having a negligible effect on the measured jet mixing. Figure 2 shows the various jet mixing and potential core regions which characterise coaxial jets. It is clear that the present experimental results are in good agreement with the Ko and Kwan study.

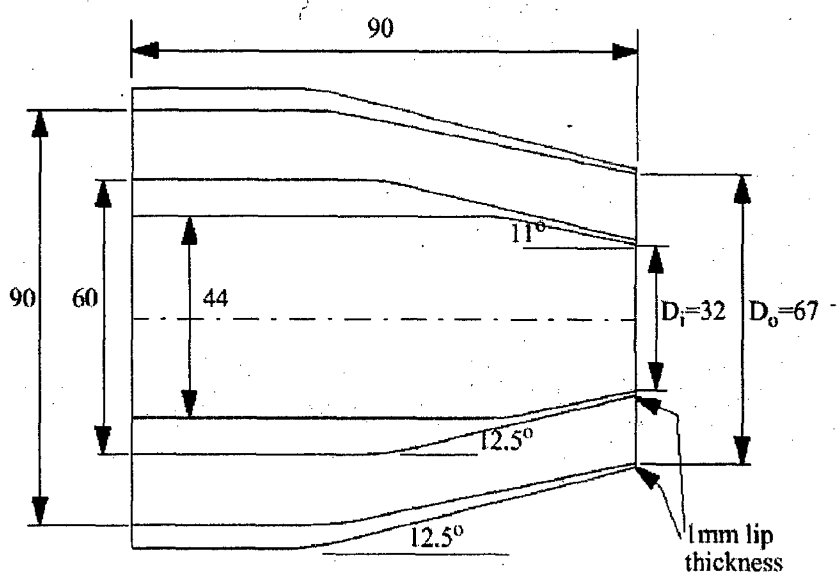


Figure 1. Co-planar, coaxial geometry

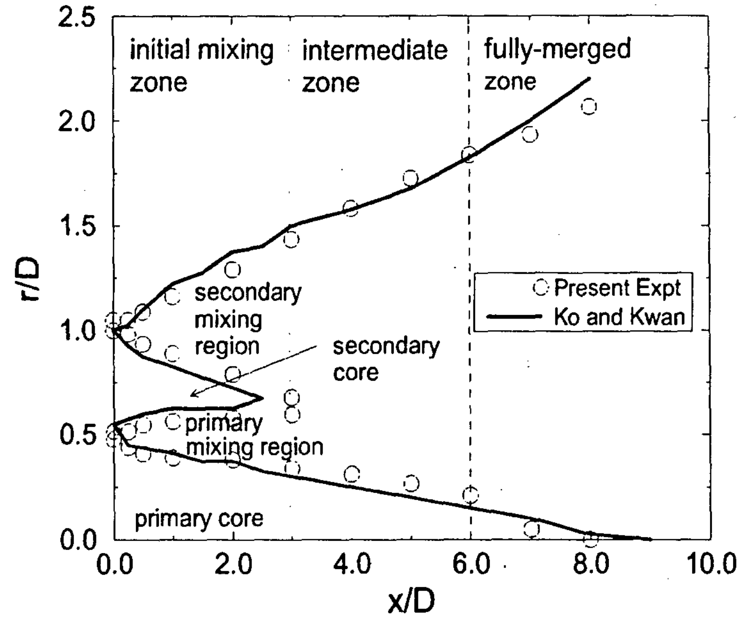


Figure 2. Coaxial jet regions

The CFD predictions used a two-dimensional axisymmetric configuration, with a downstream length of $30D_i$ as shown in Fig. 3. The grid contained four blocks with a total of approximately 10,000 grid points. This was a result of a grid refinement study and if the same grid density was used in a true three-dimensional topology would result in reasonable numbers of points. Since the experimental setup could not measure turbulence intensities and velocity profiles upstream of the nozzle, plug profiles were chosen for velocity and a sensitivity study was carried out on inlet turbulence intensity and length scale in order to achieve a reasonable match with mean and rms streamwise velocity at the first available measurement station (x/D_i=0.25), the final values used were a turbulence intensity of 3% (based on primary velocity) and a length scale of $0.5D_i$ for both the primary and secondary flows. The co-flowing stream was set at very low velocity (less than 2% of primary velocity) and itself had low turbulence levels.

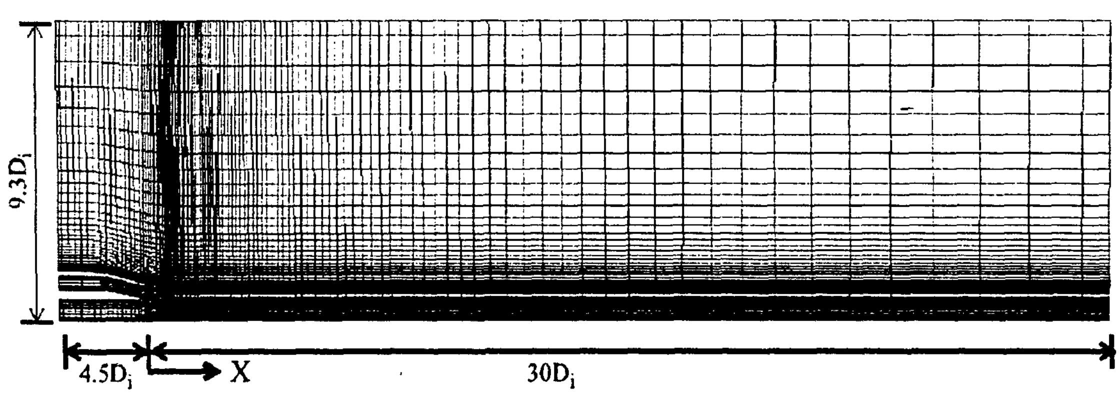
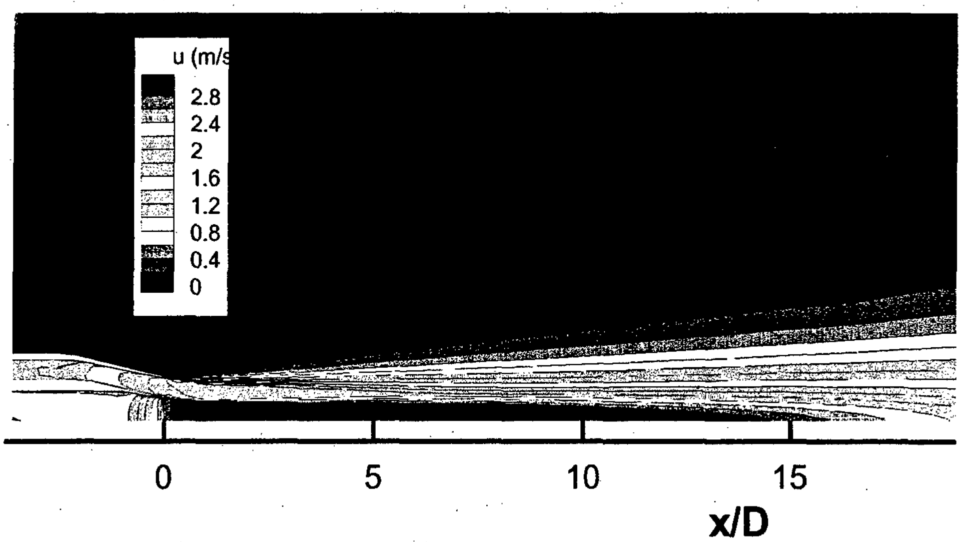
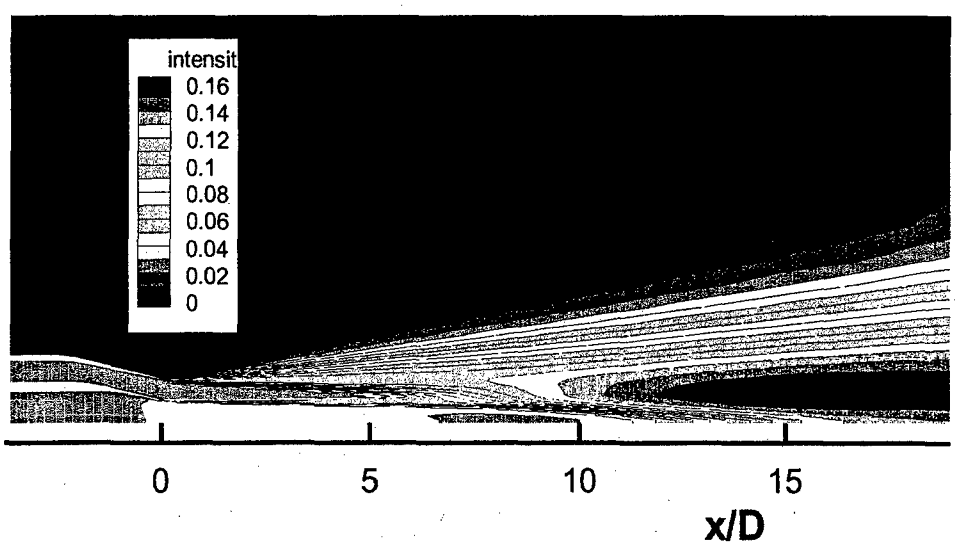


Figure 3. CFD domain and grid

Figure 4a) shows contours of axial velocity for the CFD predictions. The shear layer between the primary and secondary flow has merged into the outer shear layer by 7 primary diameters downstream, and the primary potential core length is around 12 primary nozzle diameters. The flow rates set in the experiment were such that using the geometric exit area the primary velocity should be $2.5 ms^{-1}$, however, the nozzle exit angle combined with the presence of the secondary stream causes the primary stream tube to continue contracting downstream of the exit plane giving an actual primary jet core velocity of almost $3 ms^{-1}$. As will be seen later, this is consistent with the experimental data. Contours of turbulence intensity for the CFD predictions are shown in Fig. 4b). To be consistent with the data presentation of Ko and Kwan, this is computed from the streamwise normal stress normalised by the geometric jet primary velocity. The last downstream location in the data presented by Ko and Kwan is at 8 primary diameters where the peak turbulence intensity is around 12%, the CFD predictions also show a 12% intensity at this location but the turbulence energy is *still increasing downstream* and the overall maximum is actually 16% at 17 primary diameters.



a) axial velocity



b) turbulence intensity

Figure 4. CFD prediction of calibration geometry

Comparisons of axial velocity for CFD and experiment are shown in Fig. 5, excellent agreement is observed at all axial stations. The only noticeable discrepancy is that the CFD shows a slightly 'flatter' primary velocity profile, this is to be expected since the upstream boundary conditions use a flat profile whereas the experiment has a complex feed system. Clearly this could be easily be improved by only small alterations to the upstream boundary condition profile. A comparison of turbulence intensity profiles is shown in Fig. 6. At the first location, the peak in the primary/secondary shear layer is underpredicted, but this is to be expected as the grid has only one node to resolve the peak at this station. Between 1 and 3 primary diameters downstream very good agreement is observed for both the primary/secondary shear layer peak and the primary core turbulent fluctuations. The secondary shear layer peak is underpredicted by a few % up to 3 primary diameters. At the 5 and 7 primary diameter stations the agreement is poorer with the experiment showing unexpectedly high fluctuations of around 18%. Also, it is noticeable that the experimental profiles are not smooth. Further investigation confirmed that the experimental data was taken without the presence of a co-flowing stream and low frequency unsteadiness was occurring as the jet flow spread into the experimental domain. Recent measurements have shown that this tends to increase the measured 'turbulence' levels of the furthest downstream stations by about 2%.

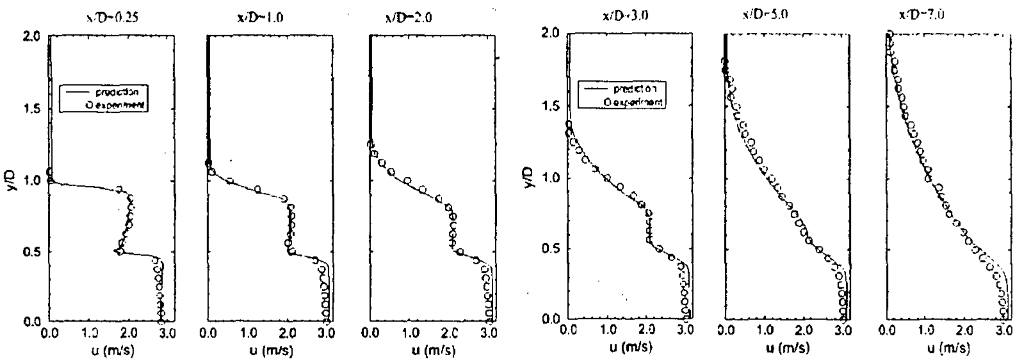


Figure 5. Mean velocity comparison, calibration geometry

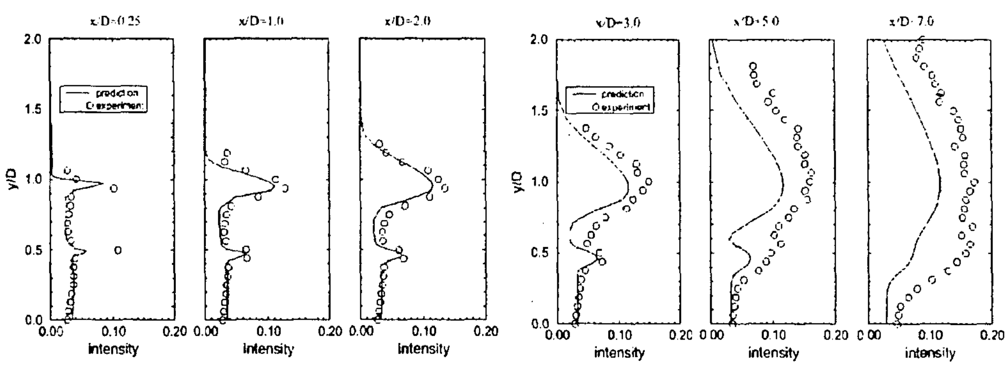


Figure 6. Turbulence intensity comparison, calibration geometry

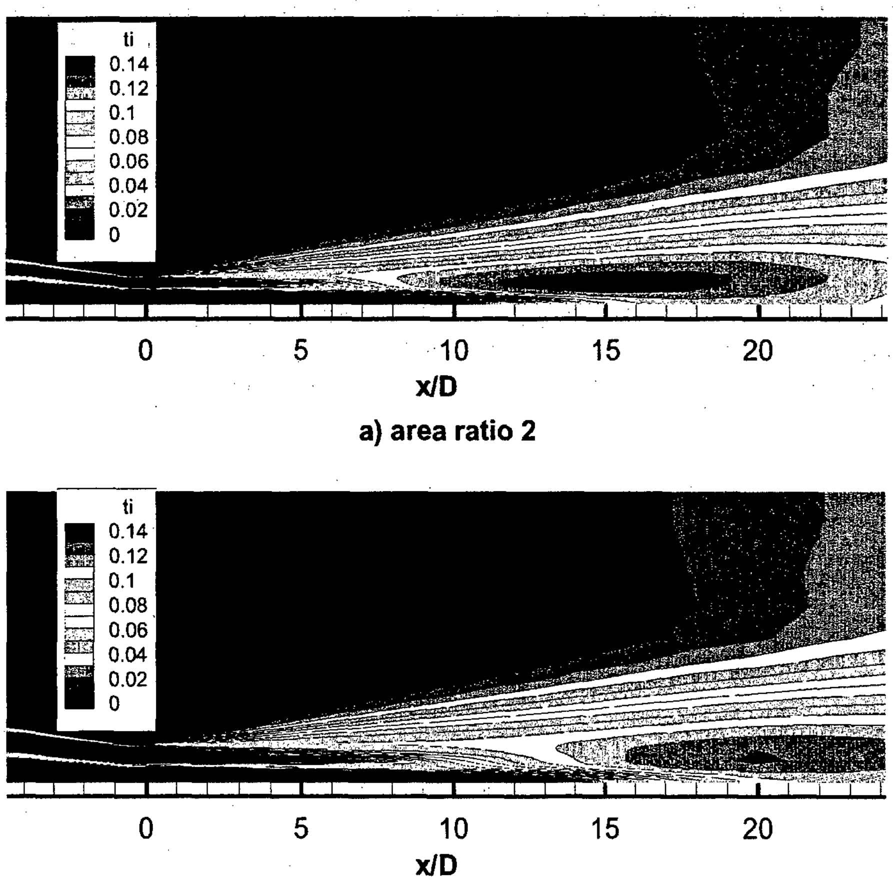
This exercise shows that the RANS-CFD methodology is capable of highly accurate velocity predictions and good turbulence predictions. Both experiment and CFD confirm that the Ko and Kwan data is reasonable for the interaction region up to 7 primary diameters, but the value of 10% intensity used in the interaction region of the four source model is at the lower bound of that found in all three data sets. The steady increase in turbulence intensity up to a peak of 16% near the end of the potential core shows that the interaction region defined in the four source model cannot be characterised by a single level of turbulence intensity. All turbulence intensity data have been defined by the primary velocity at the geometric exit, but the acceleration downstream means that the actual primary core velocity is actually 20% higher - if this were used as the reference velocity for turbulence then the CFD and experimental values quoted in this section should be factored by 0.8 in order to compare to the Ko and Kwan values. The issue of an 'effective parallel' coaxial jet is investigated further in Section 3.3.

3.2 Influence of area ratio

The four source model is calibrated using experimental data for an area ratio of 2.7, and when applied to area ratios more typical of modern turbofan engines requires adjustments in order to match experimental noise spectra. By computing the flow for two differing area ratios we can observe how the mean and turbulent quantities change and so provide a more physical justification to factors introduced into the noise model.

The geometry used in Section 3.1 causes ambiguities in that the primary flow continues to accelerate downstream of the nozzle exit. The geometry used here aims to avoid this problem by using shallower convergence angles (5° inner and 7° outer) together with a small 20mm parallel extension and has also been used in experimental noise tests. The predictions also use a longer upstream development region and upstream turbulence boundary conditions have been altered to 1% intensity (based upon primary velocity) and a length scale of 0.1 primary diameters - this was to reflect the conditions expected in the noise test facility. The Reynolds number based upon primary velocity and diameter was 500,000.

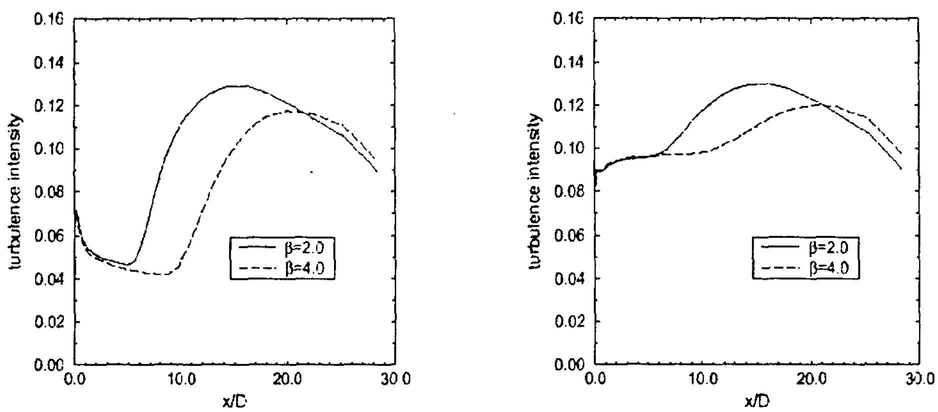
Two cases have been calculated corresponding to area ratios (β) of 2 and 4, both with a velocity ratio (λ) of 0.7. Contours of turbulence intensity (normalised by primary jet velocity) are shown in Fig. 7. The β=2 case has a shorter potential core together with a region of peak turbulence that is further upstream and of a higher magnitude. The magnitudes of turbulence intensity are more clearly seen in Fig. 8 which shows the variation along axial lines aligned with the primary and secondary nozzle lips. The primary lip line shows that the turbulence intensity in the fully mixed zone is around 1.5% higher for an area ratio of 2 as compared to 4. In the initial region, turbulence intensity is essentially identical for both area ratios gradually reducing from 7% until the secondary and primary shear layers begin to merge (at 5 and 9 primary diameters respectively) when the intensity rapidly increases. The secondary lip line (Fig. 8b) also shows little difference in the initial region with a fairly constant intensity of 9.5% until the shear layers begin to merge.



a) area ratio 2

b) area ratio 4

Figure 7. Influence of area ratio on turbulence intensity



a) primary lip line **b) secondary lip line**

Figure 8. Turbulence intensity along nozzle lip lines

The four source model in the fully mixed region uses an equivalent 'fully mixed' velocity based upon conservation of mass and momentum, this is related to the primary velocity by

$$V_m = V_i \frac{(1+\lambda^2\beta)}{(1+\lambda\beta)} \tag{1}$$

this gives for β=2, V_m=0.825V_i and β=4, V_m=0.779V_i, so that V_m for β=4 is 0.94 of V_m for β=2. If we expect that the turbulent fluctuations should scale with the fully mixed velocity, then the peak turbulence intensities should also be in this ratio, which is indeed found to be the case. This confirms that although area ratio does influence peak turbulence intensity in the fully mixed region, this can be modelled by scaling with the fully mixed velocity.

3.3 Co-planar equivalent of three-quarter length nozzle

As was shown in Section 3.1, a convergent coaxial nozzle can create accelerations in the flow downstream of the geometric nozzle exit plane. This causes difficulties with the four source model as its basis is a parallel coaxial flow with a specified primary velocity and velocity ratio. In addition, many high bypass ratio turbofan engines do not have a full length cowl (as this leads to a weight penalty), the nozzle exits are no longer co-planar and there is also the presence of the bullet. Consequently the primary and secondary flows undergo acceleration, deceleration and curvature before reaching a parallel flow which may be considered as the input to the four source model. In this section we have computed the flow in a geometry with a three quarter length cowl and integrated the flow at various stations in order to deduce equivalent jet velocity and area ratios.

Predictions were carried out in a similar manner to the earlier calculations and resolved the downstream mixing region even though this was not of immediate interest. The calculations are incompressible so that variations in density can be ignored (a more complete study would recompute this flow with the correct nozzle pressure ratios and temperatures). At the nozzle geometric exits, the nominal velocity ratio was 0.7 and the area ratio was 2.6, the co-flowing stream velocity was set at 1% of the primary velocity. Figure 9 shows contours of velocity magnitude (normalised by the primary jet exit velocity) and velocity vectors. The flow has been integrated at the three stations in order to deduce equivalent areas and velocities. The boundary between the secondary flow and the freestream (which was 0.01 of the primary) was taken as the location where

$$u = u_c + 0.05(u_o - u_c) \tag{2}$$

and the boundary between the primary and secondary as the location where

$$u = u_0 + 0.5(u_i - u_0) \tag{3}$$

and subscripts *i*, *o*, *c* refer to primary, secondary and co-flow respectively.

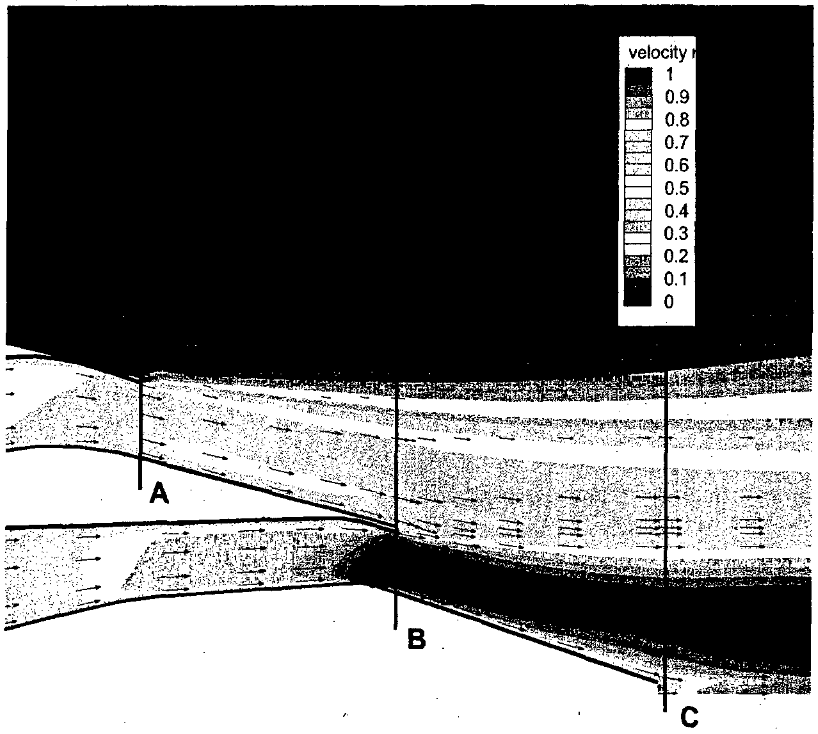


Figure 9. Three quarter length cowl velocity field

The results of the integration are given in Table 1. These show that by the primary exit plane (B), the secondary average velocity has reduced significantly, increased its mass flow due to entrainment and increased its effective area. From primary exit (B) to bullet (C) only a small deceleration is observed in both streams. The consequence is that the nominal λ=0.7, β=2.6 exit conditions are equivalent to a λ=0.5 and β=5 parallel coaxial flow immediately downstream of the nozzle. Caution should be expressed concerning the absolute accuracy of these figures as it was found that the secondary average velocity was sensitive to the location of the outer integral limit and the use of the 0.05 factor in Eq. (2) may not be the most appropriate choice.

Table 1: Integrated mean velocity and areas for three-quarter cowl

station	A	B	C
primary average velocity	-	1.0	0.97
secondary average velocity	0.7	0.49	0.45
normalised secondary mass flow	1.0	1.31	1.43
effective velocity ratio (λ)	(0.7)	0.49	0.46
effective area ratio (β)	(2.6)	4.6	5.3

4 CONCLUSIONS

Steady state Reynolds-averaged CFD predictions can be of use in providing mean flow and turbulence information for acoustic models.

When applied to coaxial jets, it is found that a CFD prediction with a k-ε turbulence gives good agreement with experimental LDV measurements for both mean velocity and turbulence intensity. The CFD methodology can then be used with confidence to assess the influence of geometric changes to nozzle design on the mean and turbulent field and hence the change in the jet noise spectra.

Computation of two coaxial jet problems with differing area ratio showed that the peak turbulence intensity is greater for the lower area ratio, but the four source model use of an equivalent 'fully mixed' velocity will correctly reproduce this effect.

Many practical turbofan have three quarter length cowls and the two streams undergo curvature and flow acceleration and deceleration before a parallel coaxial flow is achieved. A CFD prediction of this geometry showed that a velocity ratio of 0.7 and area ratio 2.6 based upon the actual nozzle exits will be similar to a parallel coaxial jet with a velocity ratio of 0.5 and an area ratio of 5.

5 ACKNOWLEDGEMENTS

This work was funded through EPSRC grant GR/M84985 in conjunction with Rolls-Royce Plc and QinetiQ. The authors are particularly grateful for the guidance given by Paul Strange and Craig Mead.

6 REFERENCES

1 Mead, C.J., Private Communication on recent work at DERA (Pyestock), 1998.

2 Mankbadi, R.R., Hayer, M.E. and Povinelli, L.A., 'Structure of Supersonic Jet Flow and Its Radiated Sound,' *AIAA Journal*, Vol. 32, no. 5, May 1994, pp897-905.

3 Mitchell, B.E., Lele, S.K and Moin, P., 'Direct Computation of Mach Wave Radiation in an Axisymmetric Supersonic Jet,' *AIAA Journal*, Vol. 35, no. 10, October 1997, pp1574-80.

4 Mankbadi, R.R., Hixon, R., Shih, S.-H. and Povinelli, L.A., 'Use of Linearized Euler Equations for Supersonic Jet Noise Prediction,' *AIAA Journal*, Vol. 36, no. 2, February 1998, pp140-47.

5 Zhao, W., Frankel, S.H. and Mongeau, L., 'Large Eddy Simulations of Sound Radiation from Subsonic Turbulent Jets,' *AIAA Journal*, Vol. 39, no. 8, August 2001, pp1469-1477.

6 Tam, C.K.W. and Auriault, L. 'Jet Mixing Noise from Fine-Scale Turbulence,' *AIAA Journal*, Vol. 37, no. 2, Feb. 1999, pp145-153.

7 Tam, C.K.W., Pastouchenko, N. and Auriault, L., 'Effects of Forward Flight on Jet Mixing Noise from Fine-Scale Turbulence,' *AIAA Journal*, Vol. 39, no. 7, July 2001, pp1261-1269.

8 Bechara, W., Lafon, P., Bailly, C. and Candel, S.M., 'Application of a k-ε Turbulence Model to the Prediction of Noise for Simple and Coaxial free jets,' *Journal of the Acoustical Society of America*, Vol. 97, no. 6, June 1995, pp3518-31.

9 Fisher, M.J., Preston, G.A. and Bryce, W.D., 'A Modelling of the Noise from Simple Coaxial Jets, Part I: with Unheated Primary Flow,' *Journal of Sound and Vibration*, Vol. 209, 1998, pp385-403.

10 Fisher, M.J., Preston, G.A. and Mead, C.J., 'A Modelling of the Noise from Simple Coaxial Jets, Part I: with Heated Primary Flow,' *Journal of Sound and Vibration*, Vol. 209, 1998, pp405-417.

11 Ko, N.W.M and Kwan, A.S.H, 'The Initial Region of Subsonic Co-axial Jets,' *Journal of Fluid Mechanics*, Vol. 73, 1976.

12 Behrouzi, P. and McGuirk, J.J., 'Experimental Studies of Tab Geometry Effects on Mixing Enhancement of an Axisymmetric Jet,' *JSME International Journal, Series B*, Vol. 41, 1998, pp908-917.

13 Jenson, L. and Menon, R.K., 'Evaluation Tests for LDV Signal Processors,' 3rd International Conference on Laser Anemometry Advances and Applications, Swansea, Wales, 1989.

14 Erdmann, J.C. and Tropea, C.D., 'Turbulence-Induced Statistical Bias in Laser Anemometry,' Proc. of 7th Biennial Symposium on Turbulence, Missouri-Roll, USA, 1981.

15 Page, G.J., Zhao, H., McGuirk, J.J., "A parallel multi-block Reynolds-averaged-Navier-Stokes method for propulsion installation applications", 12th Int. Symp. on Air Breathing Engines, Melbourne, Australia, Vol.1, 1991, pp. 864-876.

16 Launder, B.E. and Spalding, D.B. "The numerical computation of turbulent flows" *Comp.Methods in Applied Mech. and Eng.*, 3, 1974, pp. 269-289.

Reference # of Paper: 10
Discusser's Name: Dr. David J. Moorhouse
Author's Name: Dr. G. J. Page
Question:

At the end of your presentation you were commenting on actual engine parameters as opposed to an ideal experiment. Please comment on your confidence in the application of these methods to an actual full-scale engine.

Answer:

While the absolute accuracy of the CFD prediction of a complete engine may be limited, we are providing a better approximation to the four-source model and so the noise predictions should be more accurate.

Discusser's Name: Prof. Ir. Joop Slooff
Author's Name: Dr. G. J. Page
Question:

Did I understand correctly that until now you did not use the RANS data to refine the four-source model: but rather, to more precisely establish the proper values of parameters in the existing four source model?

Answer:

Initially we are using the CFD predictions to adjust parameters of the existing model. We then intend to refine the four-source model based on the CFD and LDV results.